

Please see GitHub  
link for images  
that apply to this  
document.  
Thanks!

Name: Mitchell Rice

Date: 12/5/2023

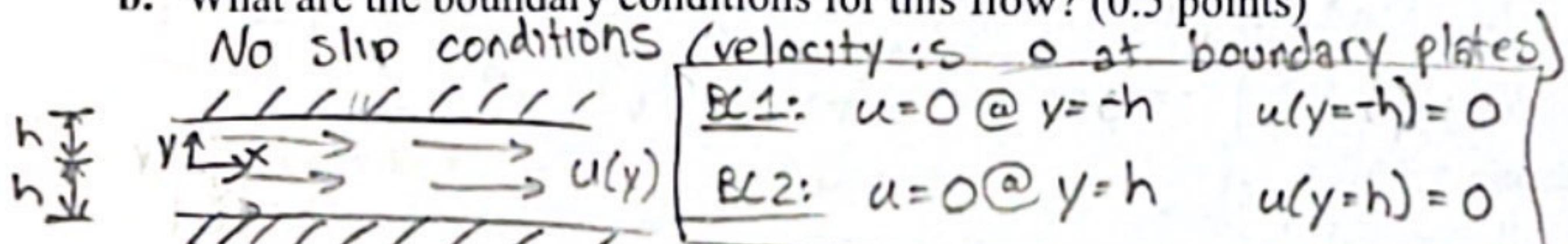
### ME 3340 Computational Fluids Bonus Assignment

- A very important part of doing any computational simulation is ensuring that your simulation is based in reality. We will practice doing this by comparing the analytical solutions for some canonical Navier Stokes flows to the computational solutions for these flows. Make sure to show your work when calculating the analytical solution. Answer on this doc or on a separate doc, save as a pdf, and submit the document renamed as "ME3340\_CFD\_YourName.pdf." Upload your COMSOL file to your github repository and include a link in your submission.

- Consider the flow between two parallel plates (see class notes). What are the assumptions that you must make about the flow? (0.5 points)

- Viscous, incompressible fluid
- Steady state, Laminar flow  $\frac{\partial u}{\partial t} = 0$
- Newtonian fluid
- Unidirectional flow  $v=w=0$
- Infinitely long and wide parallel (horizontal) plates  $\frac{\partial v}{\partial x} = \frac{\partial w}{\partial z} = 0$

- What are the boundary conditions for this flow? (0.5 points)



- Cross out the terms that reduce to zero in the appropriate continuity and Navier Stokes equations. What are the governing equations for this flow? (0.5 points)

Conservation of Mass: Infinite plates     $\cancel{v=0 \text{ unidirectional flow}}$      $\cancel{y=0}$     Infinite plates     $\cancel{\frac{\partial}{\partial z}=0}$

Steady State     $\cancel{\frac{\partial u}{\partial t} + \frac{\partial}{\partial x}(pu)} + \cancel{\frac{\partial}{\partial y}(py)} + \cancel{\frac{\partial}{\partial z}(pw)} = 0$

$0=0 \checkmark \text{Valid Flow}$

Navier-Stokes Equations:     $v=w=0$

Cartesian Equations

X:  $\cancel{\rho \left( \frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} \right)} = - \frac{\partial P}{\partial x} + \rho g_x + \mu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right)$

Steady infinite plates    unidirectional    infinite plates    your only    infinite plates    infinite plates

$\rightarrow \boxed{0 = - \frac{\partial P}{\partial x} + \mu \frac{\partial^2 u}{\partial y^2}} \quad \text{(driven by pressure)}$

Y:  $\cancel{\rho \left( \frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} \right)} = - \frac{\partial P}{\partial y} + \rho g_y + \mu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right)$

Steady infinite plates    unidirectional    unidirectional     $g_y = -g$     unidirectional    unidirectional

$\rightarrow \boxed{0 = - \frac{\partial P}{\partial y} - \rho g} \quad P = -\rho g y + f(x)$

Z:  $\cancel{\rho \left( \frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} \right)} = - \frac{\partial P}{\partial z} + \rho g_z + \mu \left( \frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right)$

Steady infinite plates    unidirectional    infinite plates    your only    unidirectional    unidirectional

$\rightarrow \boxed{0 = - \frac{\partial P}{\partial z}} \quad P = C \Rightarrow P = f(x, y)$

d. Using the boundary conditions, solve for the velocity of the fluid  $u(y)$ . (1 point)

Using governing equation for  $x$ -direction from part c.:

$$0 = -\frac{dp}{dx} + \mu \frac{d^2u}{dy^2}$$

$$\int \frac{dp}{dx} dy = \int \frac{d^2u}{dy^2} dy \quad \text{find } u$$

$$\frac{1}{2}\mu \left(\frac{dp}{dx}\right)y + C_1 = \frac{du}{dy}$$

$$\frac{du}{dy} = \frac{1}{2}\mu \left(\frac{dp}{dx}\right)y + C_1$$

$$u(y) = \frac{1}{2}\mu \left(\frac{dp}{dx}\right)y^2 + C_1 y + C_2$$

BC 1:  $u(y=0) = 0$    BC 2:  $u(y=h) = 0$

BC 1:  $u(0) = 0 = \frac{1}{2}\mu \left(\frac{dp}{dx}\right)(0)^2 + C_1(0) + C_2$

BC 2:  $u(h) = 0 = \frac{1}{2}\mu \left(\frac{dp}{dx}\right)(h)^2 + C_1(h) + C_2$

Subtract:  $0 = 0 - 2h(C_1) + 0$

$C_1 = 0$

$u(h) = 0 = \frac{1}{2}\mu \left(\frac{dp}{dx}\right)(h^2) + C_2$

$C_2 = -\frac{1}{2}\mu \left(\frac{dp}{dx}\right)(h^2)$

$u(y) = \frac{1}{2}\mu \left(\frac{dp}{dx}\right)(y^2 - h^2)$

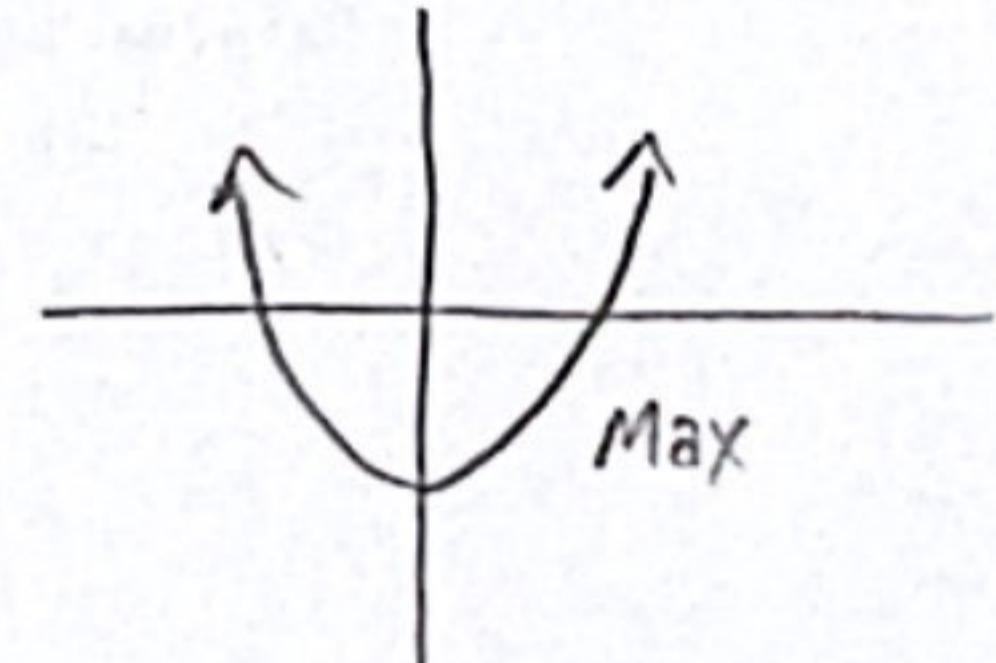
e. Calculate the maximum velocity of the fluid. (0.5 points)

We find the maximum velocity when  $(y^2 - h^2)$  is at its maximum value

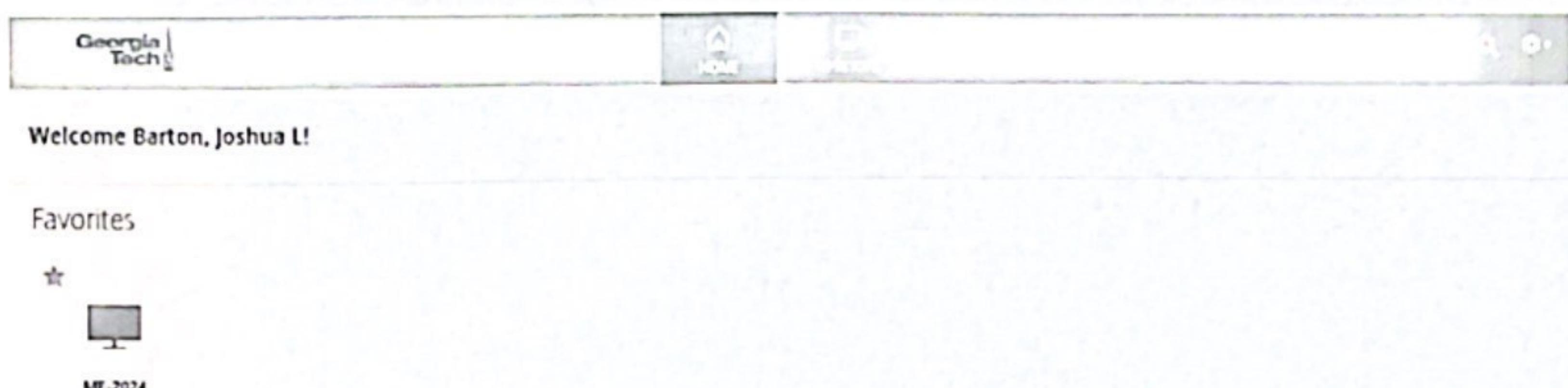
Since this is a quadratic the max magnitude occurs at the vertex  $y = -\frac{b}{2a} = \frac{0}{2} = 0$

$$u(y)_{MAX} = u(0) = \frac{1}{2}\mu \left(\frac{dp}{dx}\right)(0^2 - h^2)$$

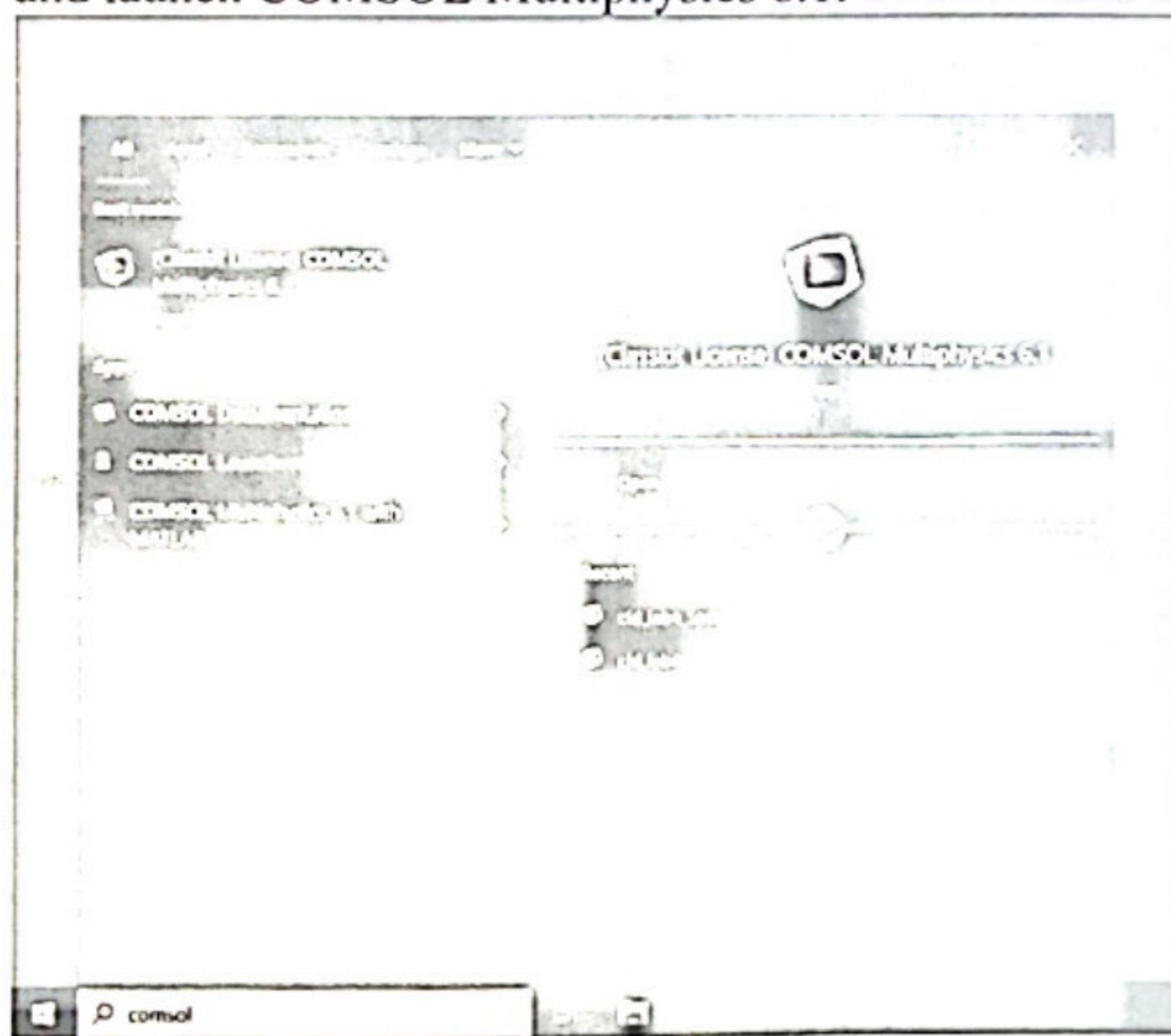
$$u(y)_{MAXMAG} = \frac{1}{2}\mu \left(\frac{dp}{dx}\right)(-h^2)$$



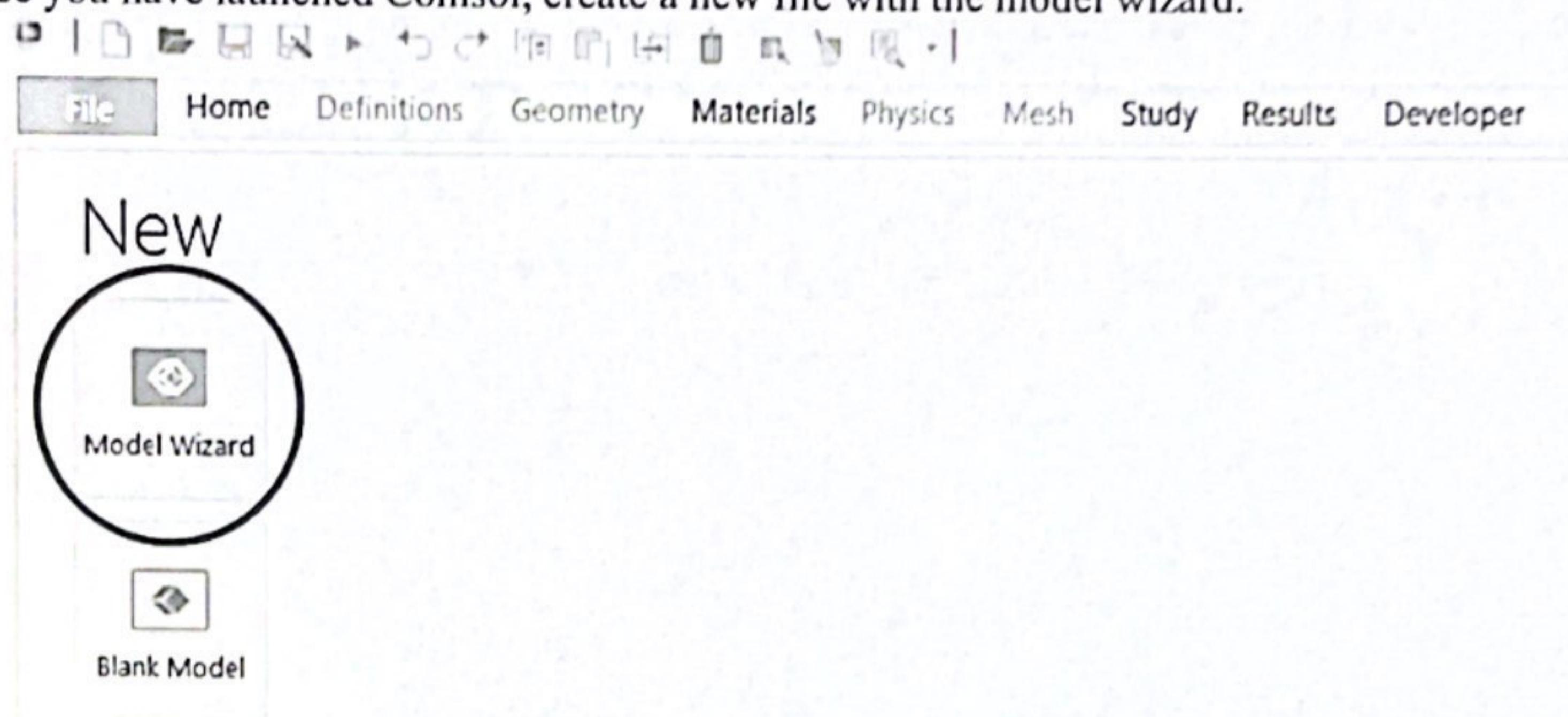
2. We now want to model this flow in a CFD software (I'll be using COMSOL Multiphysics for this). To access this software go to the Georgia Tech Vlab page (<https://mycloud.gatech.edu/>). (You may have to install Citrix Workspace for this). Your page may look something like this:



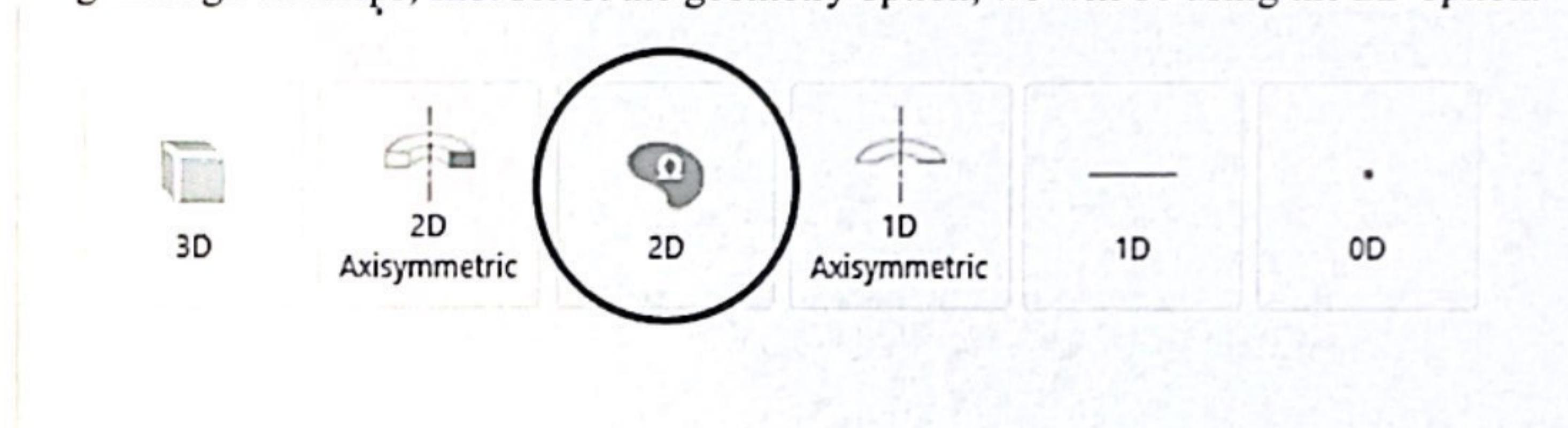
If you do not see the ME-2024 (or other year) desktop in your home folder, check the Desktops tab, and you should see some desktop denoted ME-202X. Click on the icon to launch the desktop. Once you have launched the desktop, type “Comsol” in the search bar and launch COMSOL Multiphysics 6.1.



Once you have launched Comsol, create a new file with the model wizard.



Working through the steps, first select the geometry option, we will be using the 2D option.



Now select what physics you would like to include in your study. For the purpose of this exercise we will only be using the laminar fluid flow. Select this and then click the add button and the laminar flow will be added to section at the bottom. A page will appear on the right where you can define the variable for the velocity components. Do not change these values.

## Select Physics

Search

- AC/DC
- Acoustics
- Chemical Species Transport
- Electrochemistry
- Fluid Flow
  - Single-Phase Flow
    - Creeping Flow (spf)
    - Laminar Flow (spf) **Laminar Flow (spf)**
    - Turbulent Flow
  - Rotating Machinery, Fluid Flow
    - Pipe Flow (pfl)
    - Viscoelastic Flow (vef)
    - Water Hammer (whd)
- Multiphase Flow
- Porous Media and Subsurface Flow
- Nonisothermal Flow
- High Mach Number Flow
- Particle Tracing
- Fluid-Structure Interaction
- Boundary Element

Added physics interfaces:

Laminar Flow (spf)

Add Remove

Space Dimension Study

Help Cancel Done

Review Physics Interface

Laminar Flow (spf)

Dependent Variables

Velocity field:

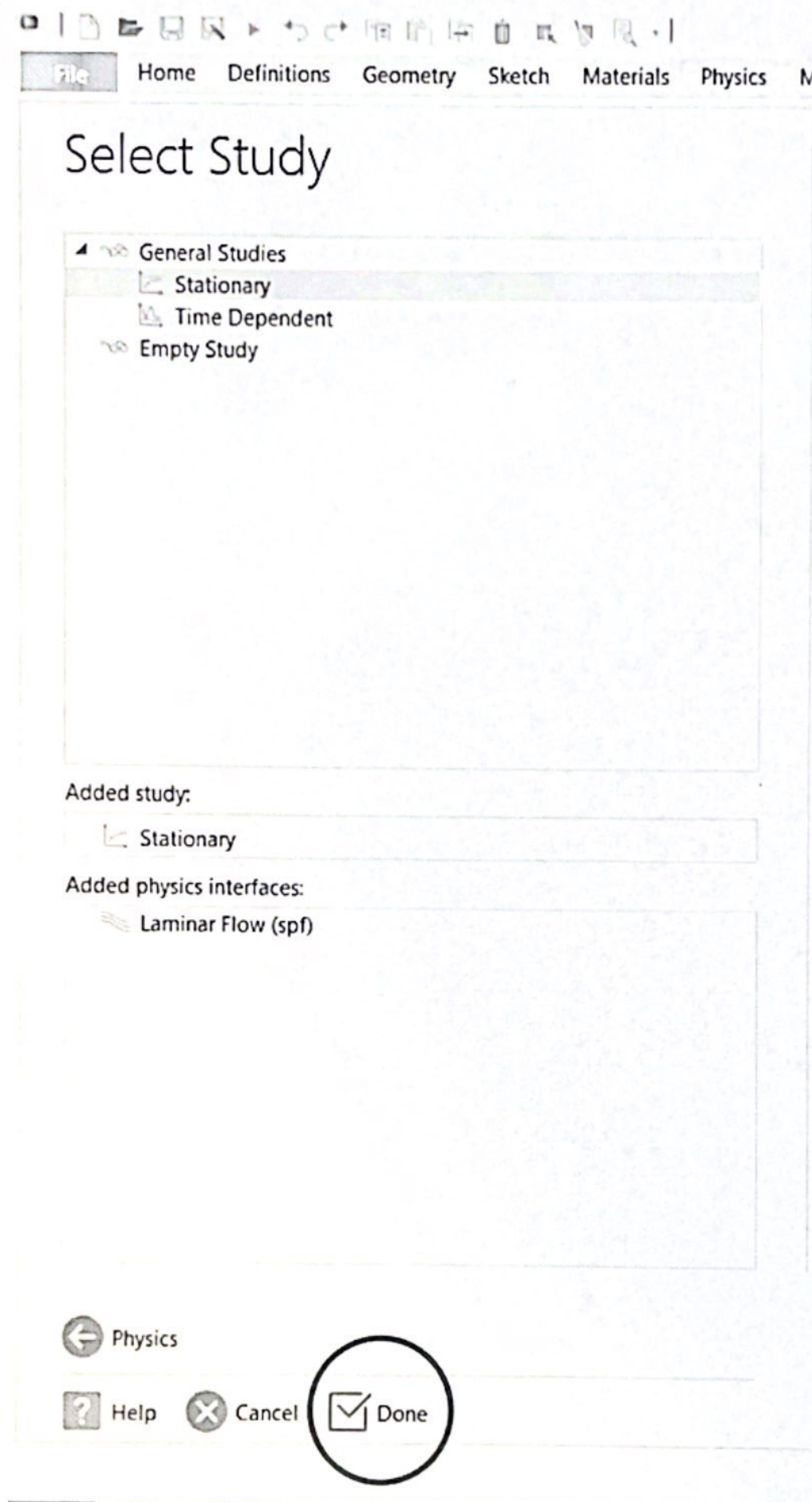
Velocity field components:

Pressure:

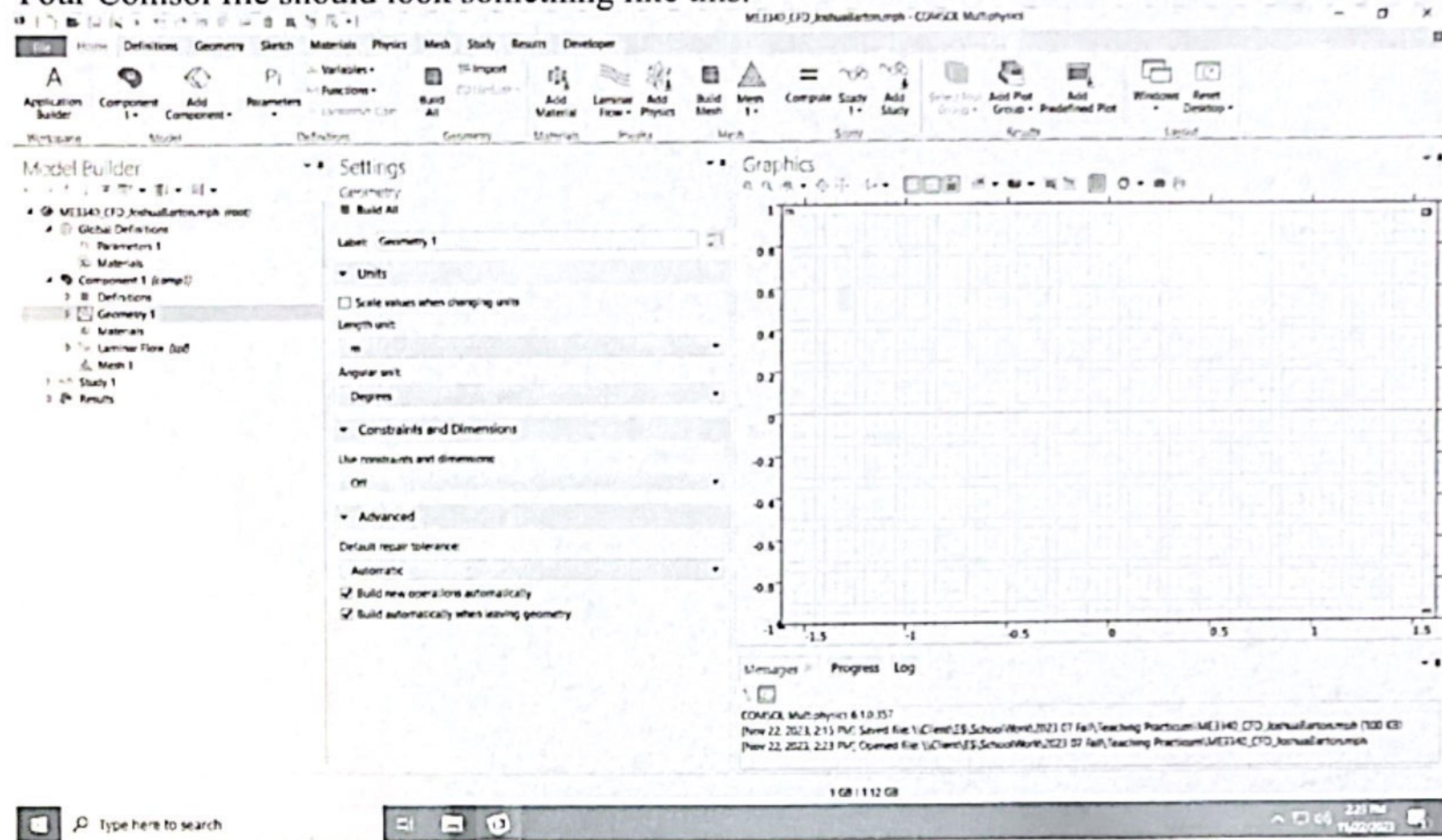
Space Dimension Study

Help Cancel Done

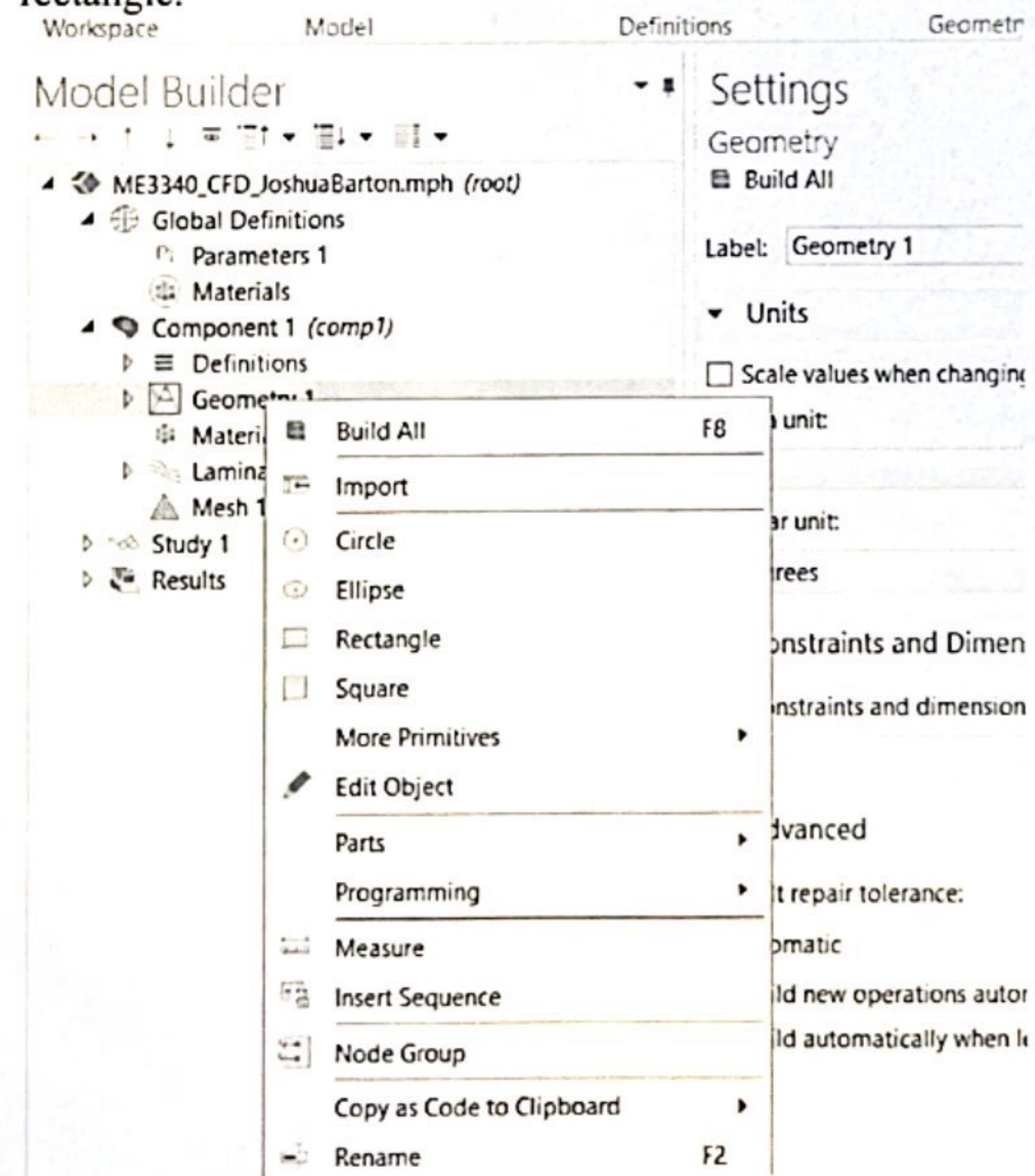
Moving to the type of study, we want to select the stationary study, and press done. Go ahead and save your CFD file as “ME3340\_CFD\_YourName.mph”



Your Comsol file should look something like this.



The first thing we must do is create the geometry, right-click the geometry section and add a rectangle.



Change the width to 0.1, the height to 0.01, and change the position type to center. Click the Build Selected button and the graphic section should display the built shape. Be wary of making your domain too large, this will make it so that turbulent flows will form in your domain and require you to use turbulent solvers, which we don't want to do for now.

### Settings

#### Rectangle

Build Selected  Build All Objects

Label: Rectangle 1

#### Object Type

Type: Solid

#### Size and Shape

Width: 0.1

Height: 0.01

#### Position

Base: **Center**

x: 0

m

y: 0

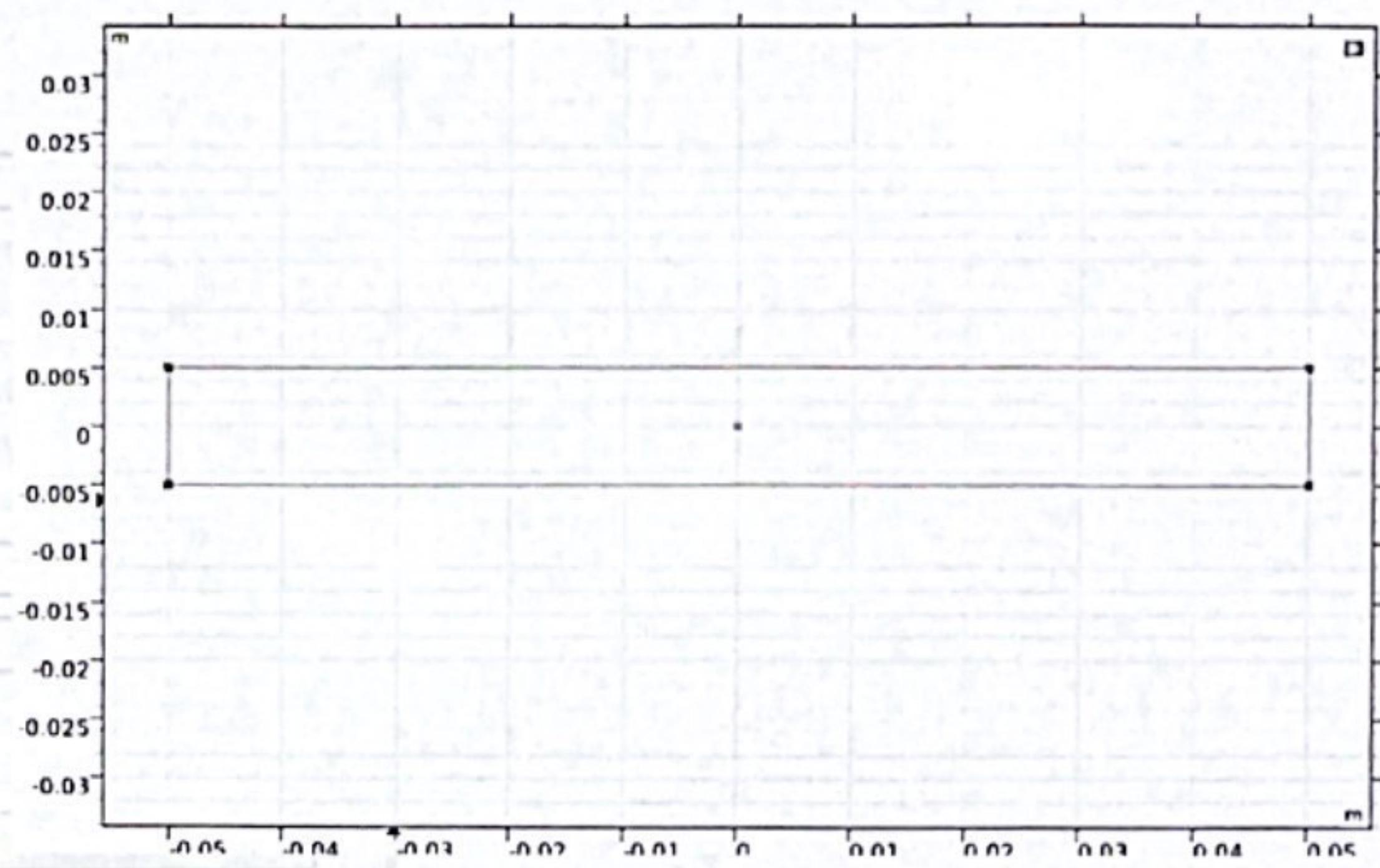
m

#### Rotation Angle

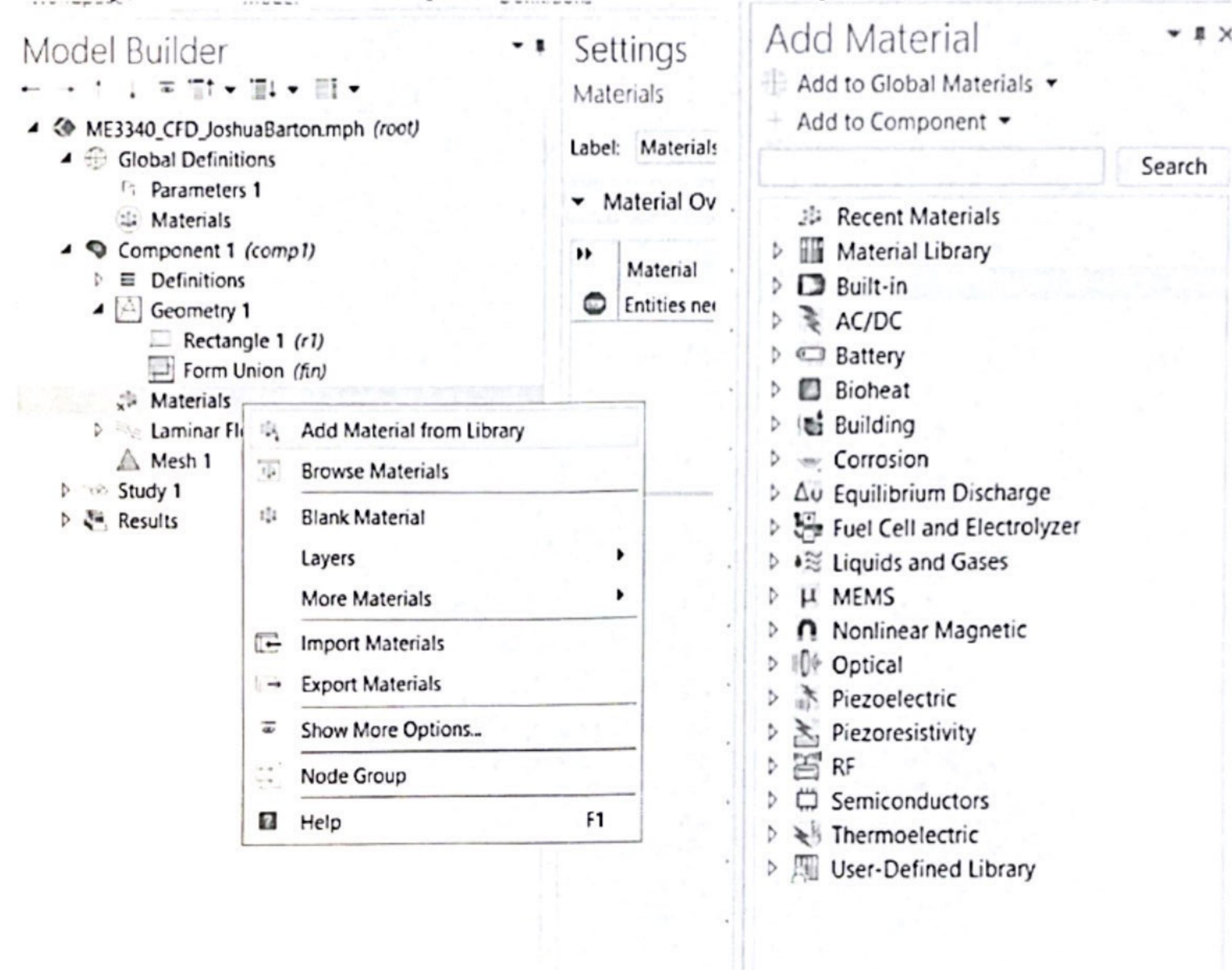
Rotation: 0

deg

#### Layers



Now assign a material to the computational domain. For this case we will select water. Right click the material section and select “Add Materials from Library.” The add material panel will come up, select the built-in dropdown, and scroll down until you see the water, liquid selection.

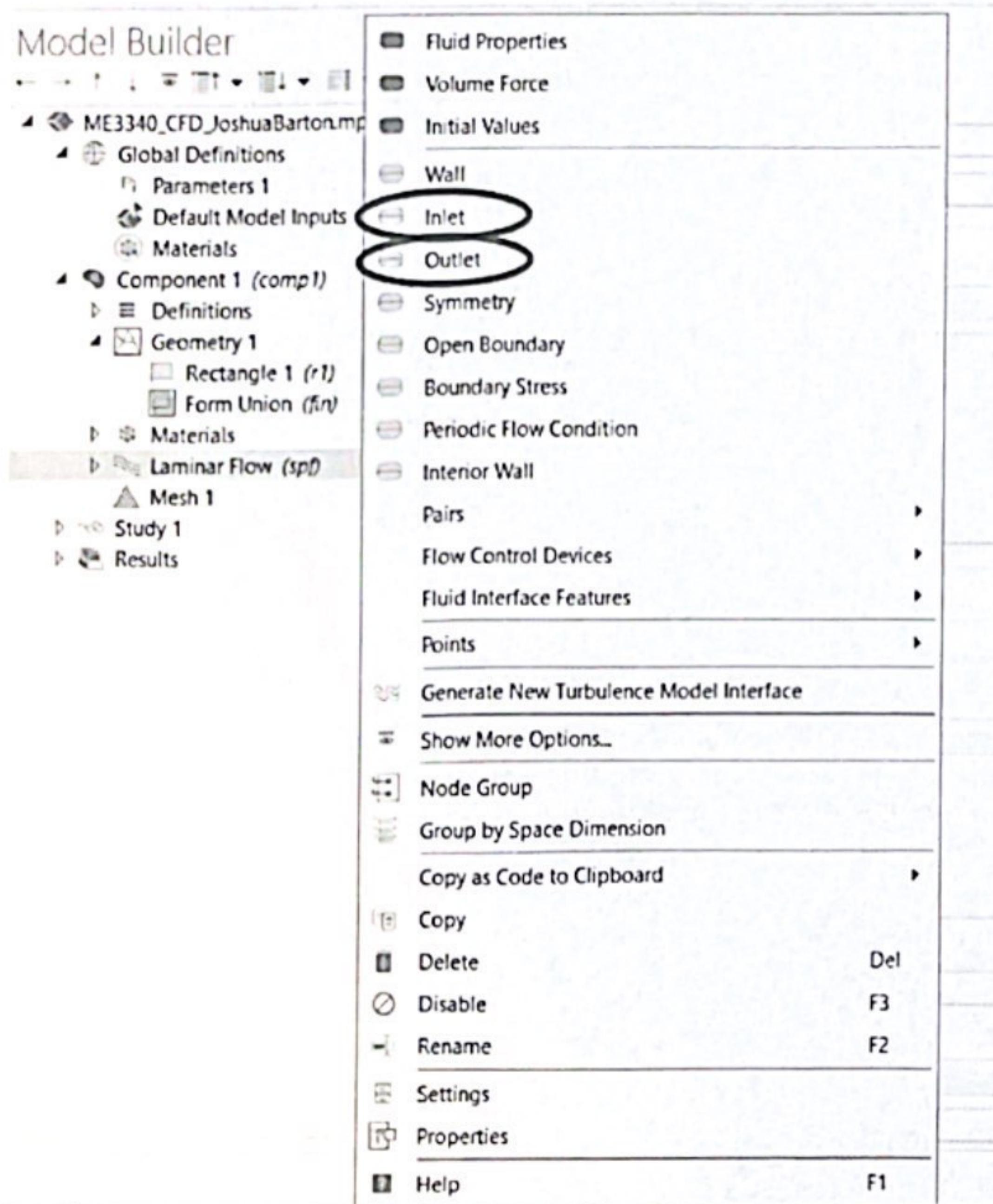


Double click the water, liquid selection.

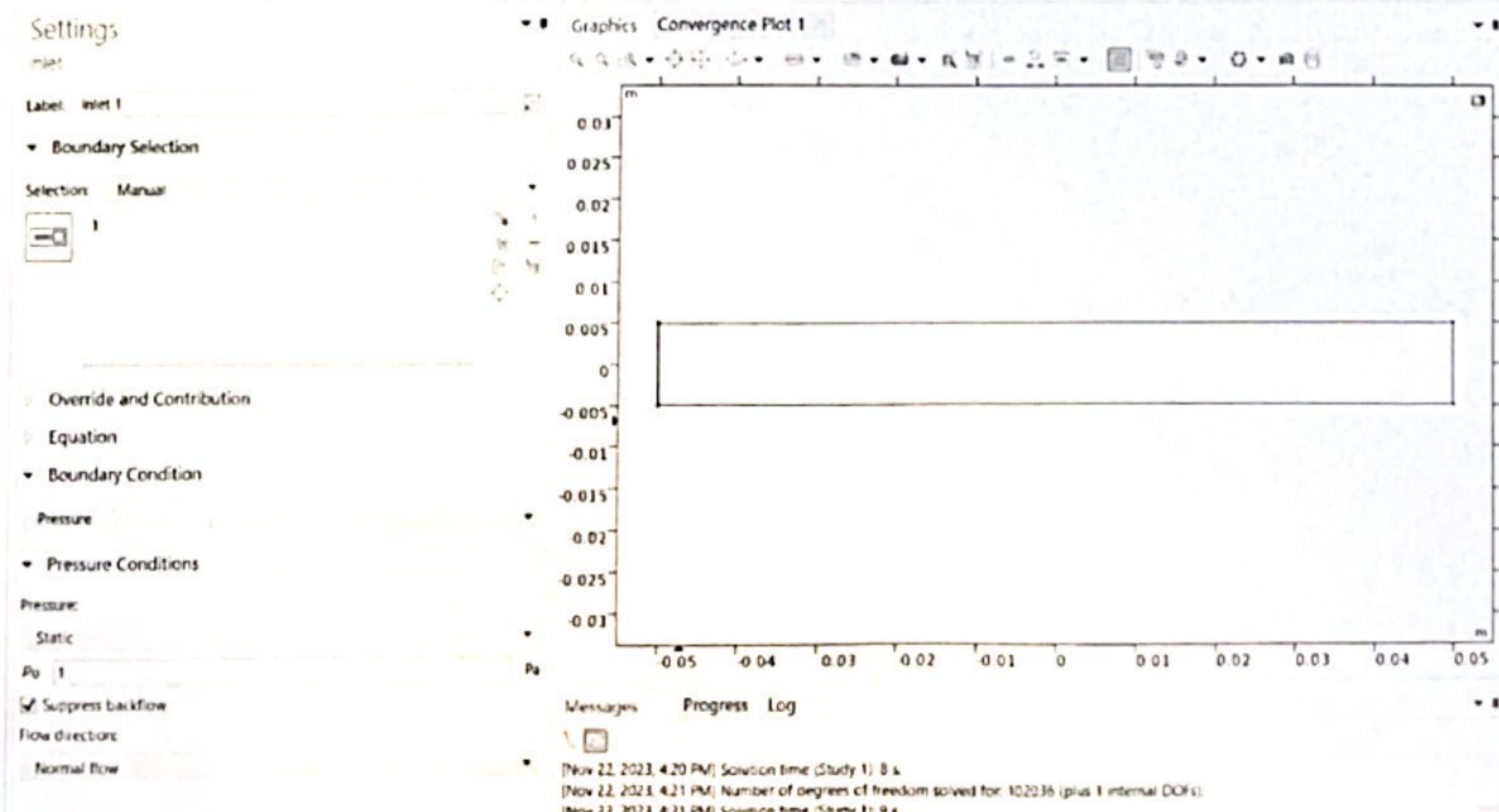
The screenshot shows two overlapping windows from a CAD application:

- Add Material Dialog:** Located on the left, it lists various material options. The "Water, liquid" option is visible in the list.
- Settings Dialog:** Located on the right, it contains the following fields:
  - Label:** Water, liquid
  - Name:** mat1
  - Geometric Entity Selection:** A section for selecting geometric entities. It includes a tree view showing a single entity labeled "1".
  - Override:** A section for overriding material properties.
  - Material Properties:** A list of properties including Basic Properties, Acoustics, Electrochemistry, Electromagnetic Models, Equilibrium Discharge, External Material Parameters, Fluid Flow, Gas Models, Geometric Properties, Magnetostrictive Models, and Piezoelectric Models.

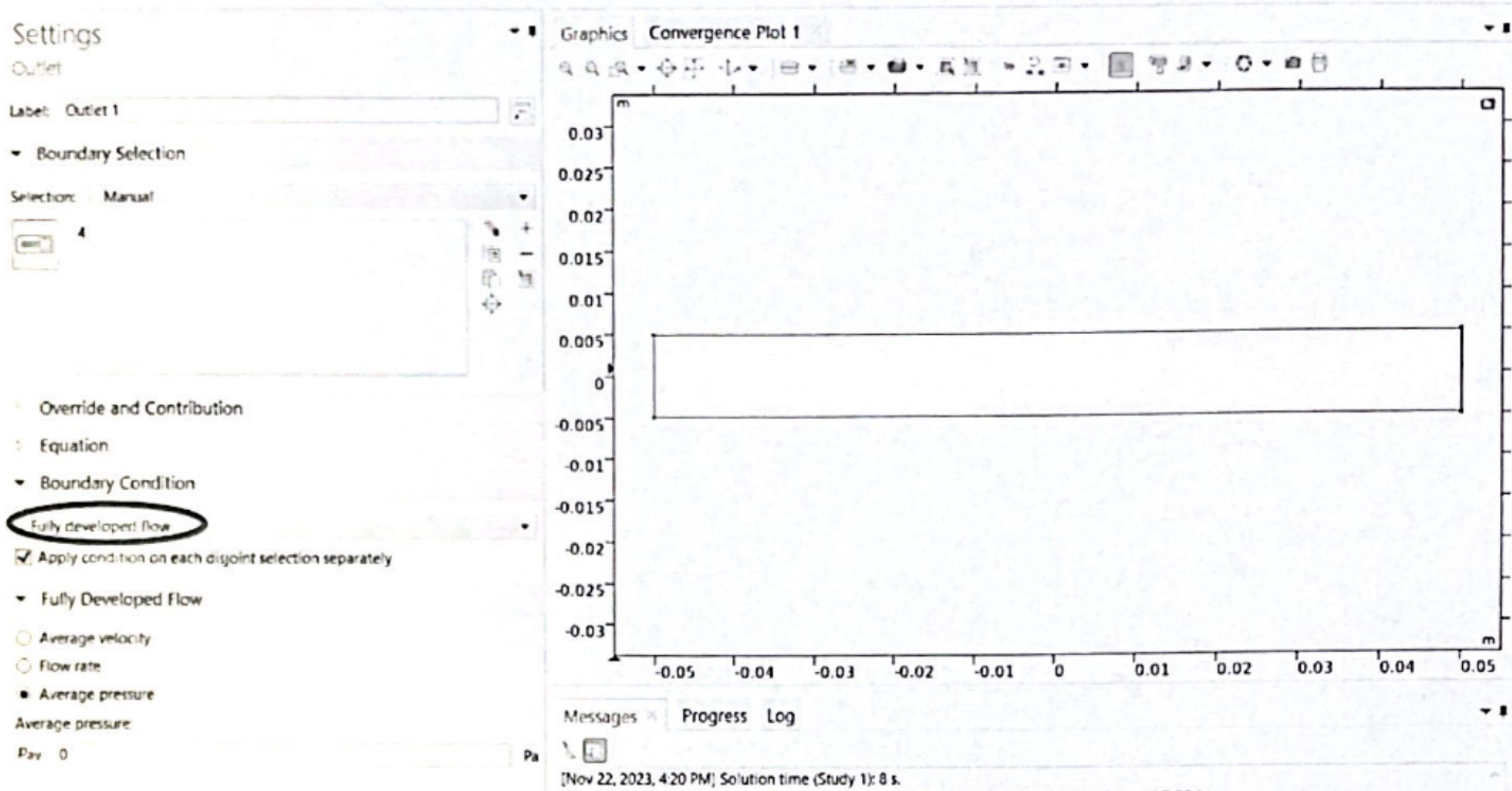
Now we need to apply the boundary conditions. Right click the Laminar Flow section and select the inlet and outlet boundary conditions. The wall boundary conditions are automatically applied.



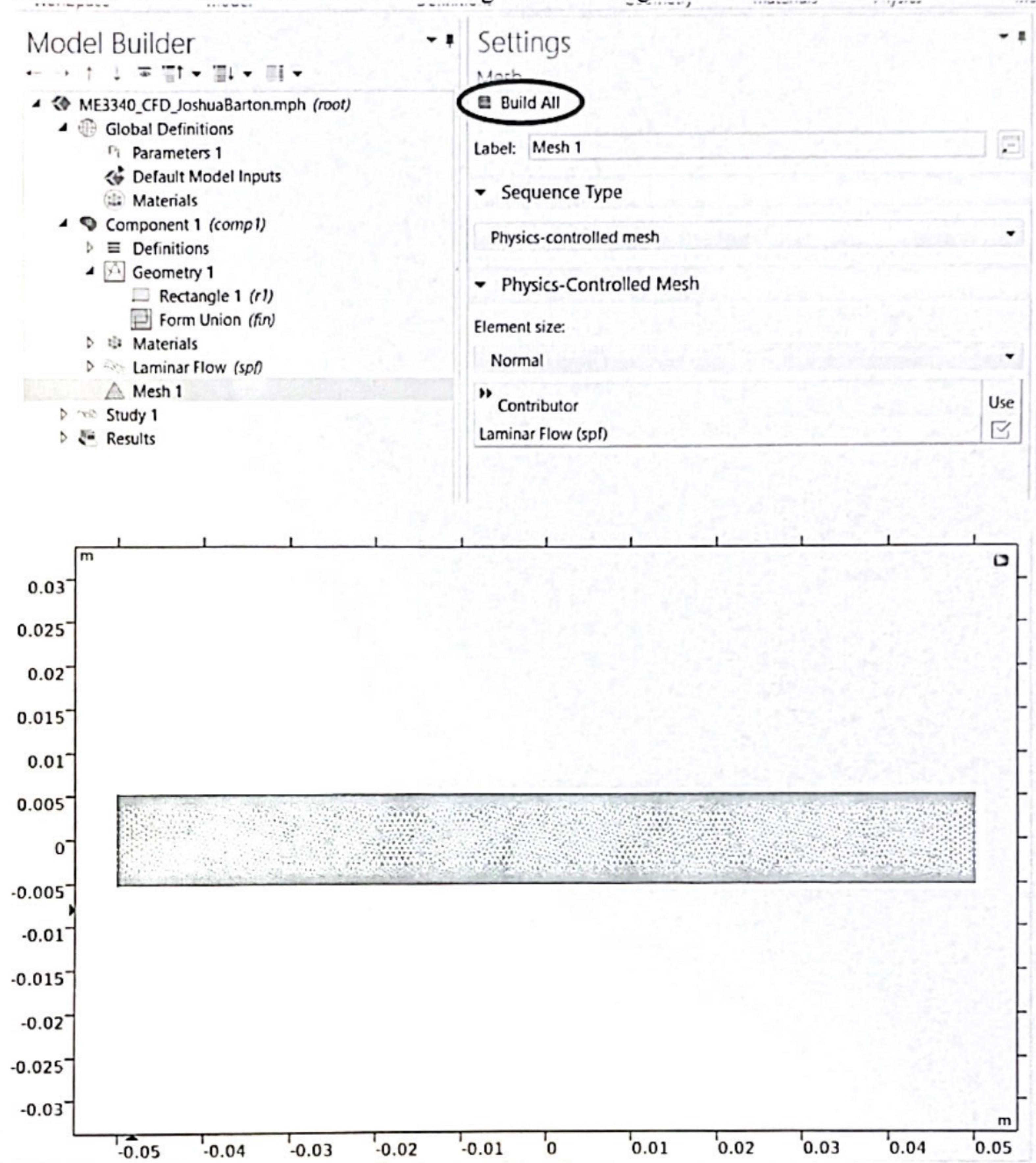
For the inlet boundary condition select the left surface, change the boundary condition to pressure and change the pressure to 1 Pa.



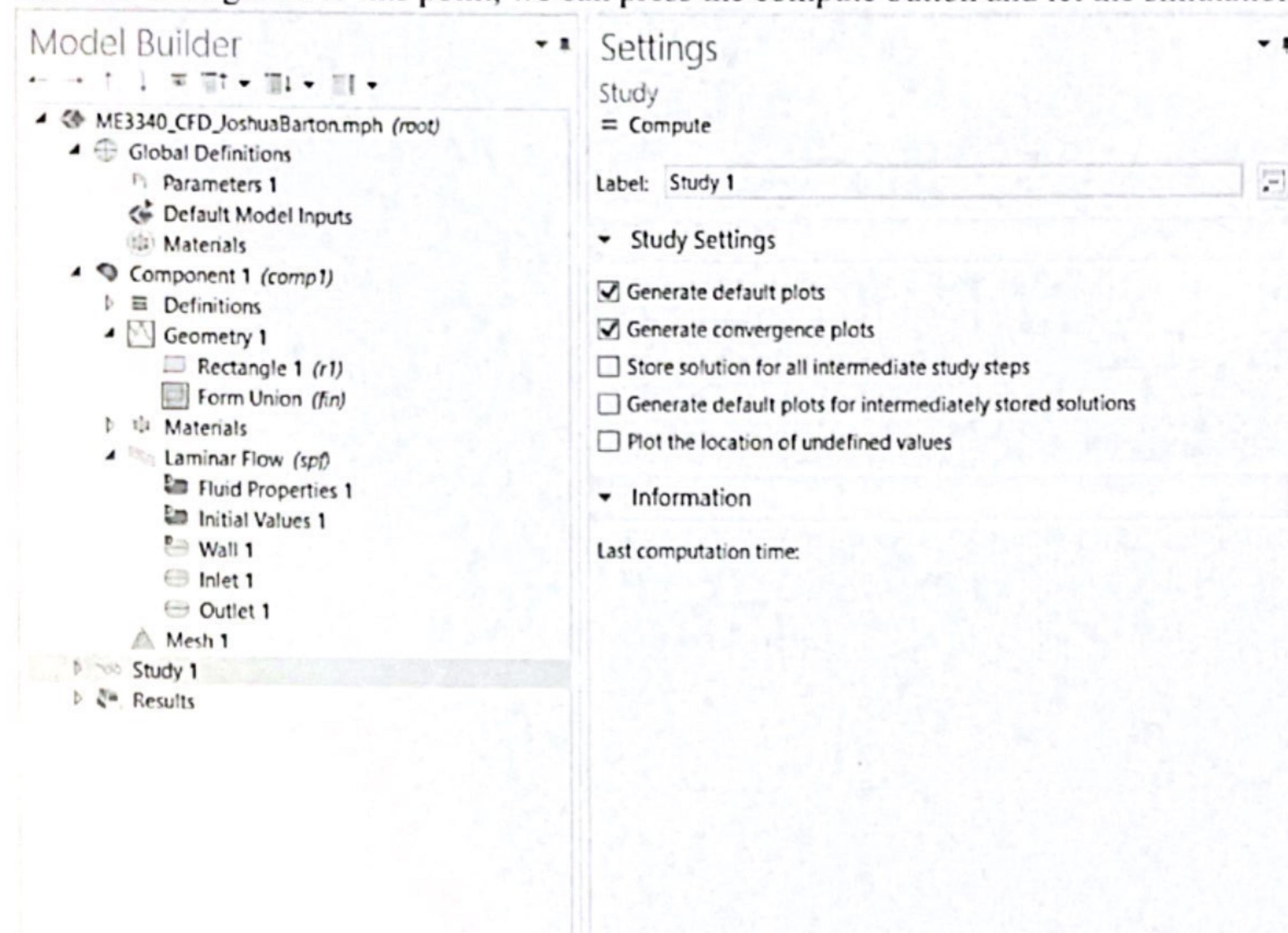
For the outlet boundary condition select the right surface, and change the type of boundary condition to fully developed flow with an average pressure of 0.



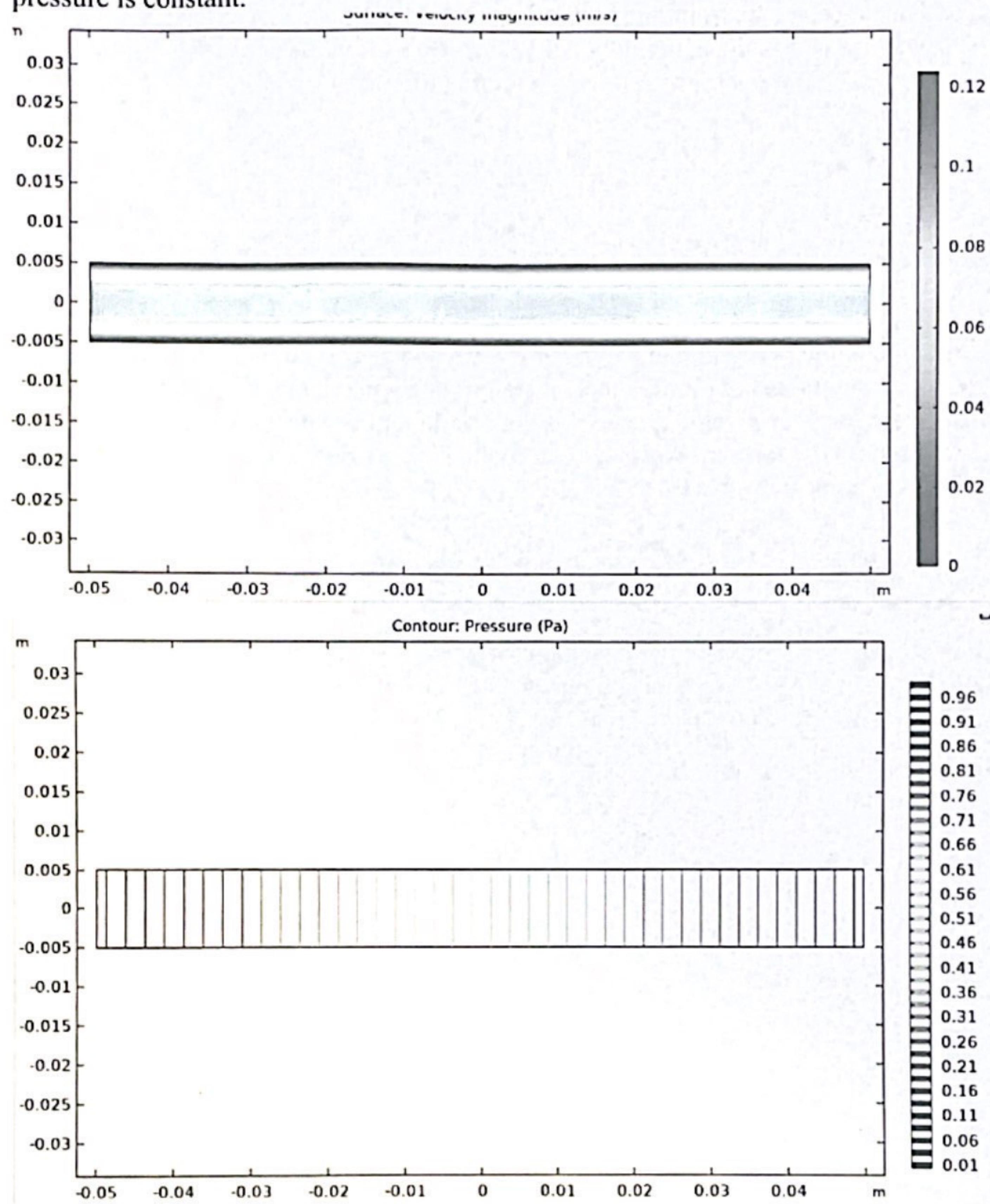
Now we need to build the mesh. We can leave the mesh on default settings, and click the build all button. The mesh should look something like this image.



Once we have gotten to this point, we can press the compute button and let the simulation run.



The default plots includes a 2D velocity plot, and a pressure plot showing lines where the pressure is constant.



Now, are these results realistic?

- a. Calculate the analytical maximum velocity for this scenario (using the pressure and distance values from our simulation, and the physical properties of water). How close is it to the computational value? (don't find the actual maximum of the computational model just analyze the plots) (1 point)

From part d. of intro,  $u(y)_{MAXMAG} = u(0) = \frac{1}{2\mu} \frac{\Delta P}{h^2}$

Pressure on left Boundary =  $190 \text{ Pa}$   $\boxed{h}$  half height of flow tube  $h = 0.005 \text{ m}$   $\boxed{\mu}$  Physical Properties  $\mu = 0.0010016 \text{ Pa}\cdot\text{s}$

Pressure on right boundary =  $0 \text{ Pa}$

$\frac{dp}{dx} = \frac{P_f - P_i}{x_f - x_i} = \frac{0 \text{ Pa} - 190 \text{ Pa}}{0.05 \text{ m} - 0.05 \text{ m}} = -190 \frac{\text{Pa}}{\text{m}} = -10 \frac{\text{Pa}}{\text{m}}$

$u_{MAX} = u(0 \text{ m}) = 2 / (0.0010016 \text{ Pa}\cdot\text{s}) (-10 \frac{\text{Pa}}{\text{m}}) / (-1) / 0.005 \text{ m}$

$u_{MAXAN} = 0.1248 \text{ m/s}$

$u_{MAX-COMP} = 0.09 \text{ m/s}$  off by about  $0.0348 \text{ m/s}$ .

- b. Plot the analytical solution of the x velocity against the computational solution (you can do this in COMSOL, right click the results section, add 1D plot group, right click the plot group and add a line plot, and select the right surface. You create expressions to plot in the x and y axis sections, remember that u is the variable for your x velocity, and y is your variable for y position). What differences do you see? Now go back to the laminar flow section and change the discretization from P1+P1 to P2+P1. What is the difference now? (2 points)

P1 + P1

Difference is,

Analytical MAX Computational MAX

$0.1248 \text{ m/s}$   $0.0955 \text{ m/s}$

Velocity Magnitude Plot ranges from 0 to  $0.09 \text{ m/s}$  contour.  
Pressure Plot has rigid non-straight lines ranging from  $-136 \times 10^{-3}$  to  $957 \times 10^{-3} \text{ Pa}$ .

P2 + P1

Difference is,

Analytical MAX Computational MAX

$0.1248 \text{ m/s}$   $0.12385 \text{ m/s}$

Velocity Magnitude Plot ranges from 0 to  $0.12 \text{ m/s}$  contour.

Pressure Plot has straight vertical lines ranging from 0.01 to 0.96 Pa.

SEE

ME3340\_EC\_(b)1V.png

ME3340\_EC\_(b)1P.png

ME3340\_EC\_(b)1L.png

(Github link)

SEE

ME3340\_EC\_(b)2V.png

ME3340\_EC\_(b)2P.png

ME3340\_EC\_(b)2L.png

(Github link)

- c. Now change the initial pressure on the inlet boundary condition. How does changing this influence the flow velocity? What happens when this value is negative? (you can use some plots to back up your explanations, but do not have to) (1 point)

I changed initial pressure to  $P_0 = 3\text{Pa}$  (from  $P_0 = 1\text{Pa}$ ). This resulted in an expected 3x increase in velocity  $u(y)_{\max} = u(0) = 0.3715$

SEE

ME3340-EC-(c)1V.png

ME3340-EC-(c)1L.png

(Github link)

I set the initial pressure to  $P_0 = -1\text{Pa}$  and the velocity fell to a maximum magnitude of  $0.0027\text{ m/s}$  and was effectively zero as shown by the plots. Backflow from negative velocity.

SEE

ME3340-EC-(c)2V.png

ME3340-EC-(c)2L.png

(Github link)

- d. Now add another wall boundary condition to the top surface and add a wall velocity to this surface. Set this velocity to  $0.1\text{ m/s}$ . How does this affect the flow? Show a 1D line plot of the velocity when you have a positive inlet pressure and a negative value (try to show the backflow). (3 points)

With wall velocity of  $0.1\text{ m/s}$  and positive inlet pressure  $P_0 = 1\text{Pa}$ :

The flow maximum velocity occurs slightly above at a value  $y > 0$ . Also, above this height, velocity approaches  $0.1\text{ m/s}$  which is the velocity of the moving wall (no slip).

SEE

ME3340-EC-(d)1V.png

ME3340-EC-(d)1L.png

(Github link)

With wall velocity of  $0.1\text{ m/s}$  and negative inlet pressure  $P_0 = -1\text{Pa}$ :

The flow velocity essentially goes to zero but there is some backflow at the top boundary shown by a negative velocity up at the moving wall shown by the 1D line plot.

SEE

ME3340-EC-(d)2V.png

ME3340-EC-(d)2L.png

(Github link)

3. Modify your input file in a new way. Explore the different types of boundary conditions or a new material. List what changes you made and if the simulation ran successfully. Why did your change influence the simulation and what are some reasons for its success if it ran and what are some reasons for its failure if it did not? (1 point)

For a new input I merged air and water for my simulation, material. Since air has a lower dynamic viscosity, the average dynamic viscosity and velocity increases.  $u(y) = \frac{1}{2\mu} \left( \frac{dp}{dx} \right) (y^2 - h^2) \rightarrow u(y) \uparrow$ .

This simulation with the air/water hybrid material ran successfully because only dynamic viscosity value changes. The velocity magnitude contour plot proves this success as velocity now ranges from 0 m/s to about 6 m/s max magnitude.

4. List anyone you worked with or any other resources that you used to help you with the assignment, if you did not receive any assistance with the assignment please put N/A to receive the point for this question, do not leave it blank. (1 point)

N/A